

A vertex-based finite-volume algorithm for the Navier-Stokes equations

S. K. Chakrabartty and K. Dhanalakshmi

Computational and Theoretical Fluid Dynamics Division
National Aerospace Laboratories, Bangalore 560 017, INDIA

Abstract: A vertex-based, finite-volume algorithm has been developed to solve the Reynolds-averaged Navier-Stokes equations without thin-layer approximation. An explicit, five-stage Runge-Kutta, time-stepping scheme has been used for time integration along with different acceleration techniques to reach the steady state. A code employing multi-block grid structure has been developed. This code can accept any type of grid topology. As test cases, the turbulent flow past RAE-2822 and NACA-0012 airfoils, and the laminar flow past a cropped delta wing at ten degrees angle of attack have been computed and the results compared with available numerical and experimental results. The Baldwin-Lomax turbulence model has been used in the case of turbulent flows.

1. Introduction

The unsteady, three-dimensional Navier-Stokes equations, describing the conservation of mass, momentum, and energy, form a generally accepted set of governing equations to model a continuum flowfield. For laminar flows, this system is closed once the constitutive laws and shear stress tensor relations are defined. The hybrid character of the equations and their non-linearity make it impossible to solve them analytically except for a few simple cases. With modern high speed computers and robust numerical algorithms, it is now possible to get a numerical solution of a problem of aerodynamic interest within a reasonable time and with a high degree of accuracy.

For the spatial discretization of the flow variables, finite-difference, finite-element, and finite-volume methods have contributed a lot to the improvement of the accuracy and convergence of numerical schemes and have been extensively and successfully tested for the Euler and Navier-Stokes equations for complex geometries. Of these three methods, the finite-volume method has the advantage of simulating the derivatives more accurately in physical space using Green's theorem over a control volume in highly stretched and skewed grids. Finite-volume methods are of two basic types depending on whether the flow variables are defined at the centres of the cells (cell centred) or at the vertices of the cell (cell vertex/vertex based). Vertex-based finite-volume schemes are more accurate than the cell-centred schemes in approximating the derivatives in a highly stretched and skewed grids. This has been shown analytically by Chakrabartty (1990a) for the two-dimensional case and the analysis holds true for the three-dimensional case also. Another advantage of using vertex-based schemes is that, since the vertices of the cells lie on the solid surface for the boundary-fitted grid, the pressure can be calculated directly on the surface. On the other hand, for cell-centred schemes, the surface pressure is obtained using either an extrapolation or a normal momentum relation with the boundary-layer type of approximation, i.e., the normal derivative of pressure is zero inside the boundary layer. In order to reduce computer memory and CPU time requirements, some researchers use thin-layer approximations for both two- and three-dimensional flow computations (Chakrabartty

1989, 1990b, 1991, Radespiel 1989, Swanson and Turkel 1985, Kumar 1989). A novel all-vertex finite-volume method for two-dimensional flow has been presented by Chakrabartty (1990a). This method has the property that whether or not one uses the thin-layer approximation, it needs almost same CPU time and memory.

In this report, the method described by Chakrabartty (1990a) for solution of two-dimensional Navier-Stokes equations with a modified calculation of the time-step limit has been applied to cases with shocks and also extended to three-dimensional flows.

2. Methodology

The Reynolds-averaged Navier-Stokes equations have been considered here in integral form along with the constitutive relations and the equation of state. The details of the equations, finite-volume formulation, boundary conditions, time stepping scheme and the stability conditions have been discussed by Chakrabartty and Dhanalakshmi (1993).

An artificial dissipation term has been added to damp the high frequency oscillations in the major portion of the flowfield where the viscous effects are small.

Local time stepping, enthalpy damping and residual smoothing have been used to accelerate the rate of convergence to steady state.

The algebraic turbulence model of Baldwin and Lomax (Baldwin and Lomax 1978, Chakrabartty 1990b) has been used here for the turbulent flow cases.

A C-type grid based on the direct algebraic method of Jain (1983) has been generated for the airfoils and for the three-dimensional case an O-O grid is generated using transfinite interpolation method (Kumar 1987).

3. Results and discussion

3.1. Two-dimensional flows

Two test cases have been considered in the present study - (a) the turbulent transonic flow past the NACA-0012 airfoil at a freestream Mach number $M_\infty = 0.799$, an angle of attack $\alpha = 2.26^\circ$ and a freestream Reynolds number $Re = 9 \times 10^6$, and (b) the turbulent transonic flow past the RAE-2822 airfoil at $M_\infty = 0.73$, $\alpha = 2.79^\circ$ and $Re = 6.5 \times 10^6$. The computed surface pressure distributions for these cases are compared with the corresponding experimental results (Cook *et al.* 1979, Harris 1981) in Figures 1 and 2. The comparison is extremely good in the case of RAE-2822 airfoil. For the NACA-0012 airfoil, due to the existence of a strong shock and shock induced separation, the algebraic turbulence model does not give a satisfactory comparison. Better turbulence models are under study to handle such cases.

3.2. Three-dimensional flows

For this case, the flow past a 65° swept cropped delta wing with rounded leading edge at $M_\infty = 0.85$, $\alpha = 10^\circ$ and $Re = 2.38 \times 10^6$ has been considered. This configuration has also been considered by Rizzi and Müller (1989). In their computations a fine grid has been used and the computed results have been compared with experimental results. They have found that their results compare well with experiment for both turbulent and laminar flow conditions. So this is a good test case to validate a laminar flow computational code. In the present case, computations have been performed on two grids - one of size $65 \times 25 \times 33$ and another of size $129 \times 49 \times 65$. The two grids have been considered to study the dependence of the computed solution on the grid. Lines of constant C_p

on the lower and upper surface of the wing have been shown in Fig. 3 and the skin-friction lines on the upper surface are shown in Fig. 4 along with the same obtained by experiment (Bannick and Houtman 1986). Looking along the flow direction, a diverging pattern indicates reattachment and converging pattern shows separation. The primary and secondary separation and the reattachment lines show good qualitative agreement with experiments. Streamlines emanating from the leading edge and extending upto the wake region have been shown in Fig. 5. For a quantitative comparison with experiments, the spanwise pressure distributions obtained from both crude and fine grid computations at three chordwise locations, $x = 0.30, 0.60$ and 0.80 , have been shown in Fig. 6 along with the same obtained by Euler calculations. The improvement of the Navier-Stokes calculations over the Euler calculations is apparent from the figures. The fine grid results show a marked improvement over the crude grid solution. Although the fine grid computation is expensive in our present computing facility, the improvement shown in the results due to grid refinement is encouraging. The comparison of the total force coefficients also shows good overall agreement with the experiment.

4. Conclusions

The results obtained for two-dimensional flows are quite good. The turbulence model needs to be improved to handle difficult problems such as transonic flows with strong shocks and turbulent shock-induced separation. A more accurate model may give a better comparison with experiment in the case of the NACA-0012 airfoil. Regarding three-dimensional flows, for better accuracy of the results, two issues need to be addressed - (a) grid fineness and (b) turbulence modelling. With the available computing facilities at NAL, it is difficult to perform computations on very fine grids. The $65 \times 25 \times 33$ -grid computation required about 2 min. of CPU time/iteration while, the $129 \times 49 \times 65$ -grid computation required 20 MB of memory and about 18 min. of CPU time/iteration on a Magnum Multi-RISC computer. In order to obtain a converged solution, the code needed about a thousand iterations. The number of iterations depends on the complexity of the flow and the desired accuracy. Our attention is more towards a better grid quality in terms of the topology and orientation, such that, with not-so-fine a grid, better solution can be achieved using the limited computing resources. The extension of this turbulence model to three dimensions is in progress and at the same time we are looking for other more accurate models in the literature.

Acknowledgements

During the development of the present code, the help and guidance received from a three-dimensional Euler code developed by Dr. Anand Kumar is gratefully acknowledged. The grid generation package and the post-processing routines used in the present work have also been developed by Dr. Anand Kumar and his colleagues in the CTFD Division. The authors take this opportunity to thank Dr. S.S. Desai, Head, CTFD Division for his continuous encouragement and support for this work. The success of this project will be a fitting tribute to Prof.R. Narasimha, Director, NAL, under whose encouragement and continuous guidance this project was initiated and is proceeding smoothly.

References

- Baldwin, B.S. and Lomax, H. (1978), "Thin layer approximation and algebraic model for separated turbulent flows," AIAA Paper No. 78-257
- Bannick, W.J. and Houtman, E.M. (1986), "Experiments on the transonic flow over a delta wing at high angles of attack," *Proc. of the International Vortex Flow Experiment*

on *Euler Code Validation*, Aeronautical Research Institute of Sweden, Stockholm, FF/TN 37-46.

Chakrabartty, S.K. (1989), "Numerical solution of Navier-Stokes equations for two dimensional viscous compressible flows," *AIAA X*, 27(7), pp.843-844.

Chakrabartty, S.K. (1990a), "A finite volume nodal point scheme for solving two dimensional Navier-Stokes equations," *Ada Mechanica*, 84, pp. 139-153.

Chakrabartty, S.K. (1990b), "Vertex based finite volume solution of the two dimensional Navier-Stokes equations," *AIAA Journal*, 28(10), pp.1829-1831.

Chakrabartty, S.K. (1991), "Some results of compressible viscous flows past airfoils using a modified time step for stability," NAL PD CF-9129.

Chakrabartty, S.K. and Dhanalakshmi, K. (1993), "A vertex based finite volume algorithm for three dimensional Navier-Stokes equations," NAL PD CF-9243.

Cook, P.H., McDonald, M.A., and Firmin, M.C.P. (1979), "airfoil RAE 2822 - pressure distributions and boundary layer and wake measurements," AGARD AR-138.

Harris, C.D. (1981), "Two-dimensional aerodynamic characteristics of the NACA ()12 airfoil in the Langley 8-foot transonic pressure tunnel," NASA TM-81927.

Jain, R.K. (1983), "Grid generation about an airfoil by an algebraic equation method," DFVLR-IB-129-83/40.

Kumar, A. (1987), "Application of transfinite interpolation to grid generation," *Proc. of Third National Conference on Aerodynamics*, Bombay, 27-28, November, 1987.

Kumar, A. (1989), "Flow calculation over delta-wing using thin-layer Navier-Stokes equations," *Proc. of the International Symposium on Computational Fluid Dynamics*, Nagaya, August 28-31, pp.342-347.

Radespiel, R. (1989), "A cell-vertex multi-grid method for the Navier-Stokes equations," NASA TM-101557.

Swanson, R.C. and Turkel, E. (1985), "A multistage time stepping scheme for the Navier-Stokes equations," AIAA Paper No. 85-0035.

Rizzi, A., and Müller, B. (1989), "Large-scale viscous simulation of laminar vortex flow over a delta wing," *AIAA J.*, 27(7), pp.833-840.

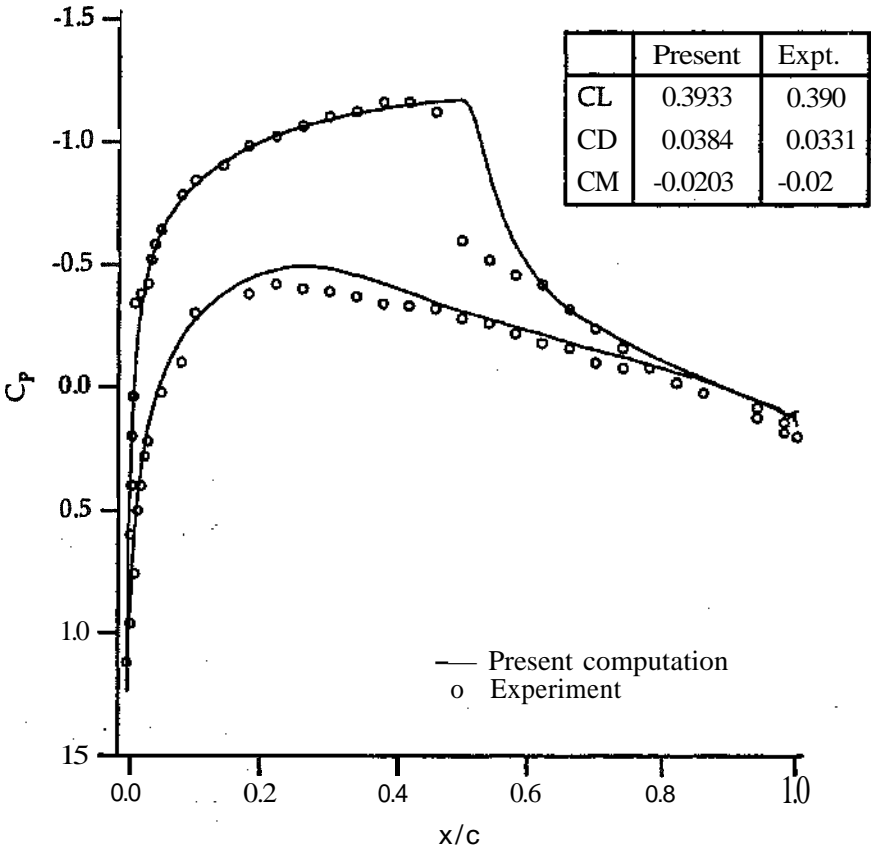


Figure 1. Surface pressure distribution for a NACA-0012 airfoil.
($M_\infty = 0.799$, $\alpha = 2.26^\circ$, $Re = 9 \times 10^6$, Grid = 257×62)

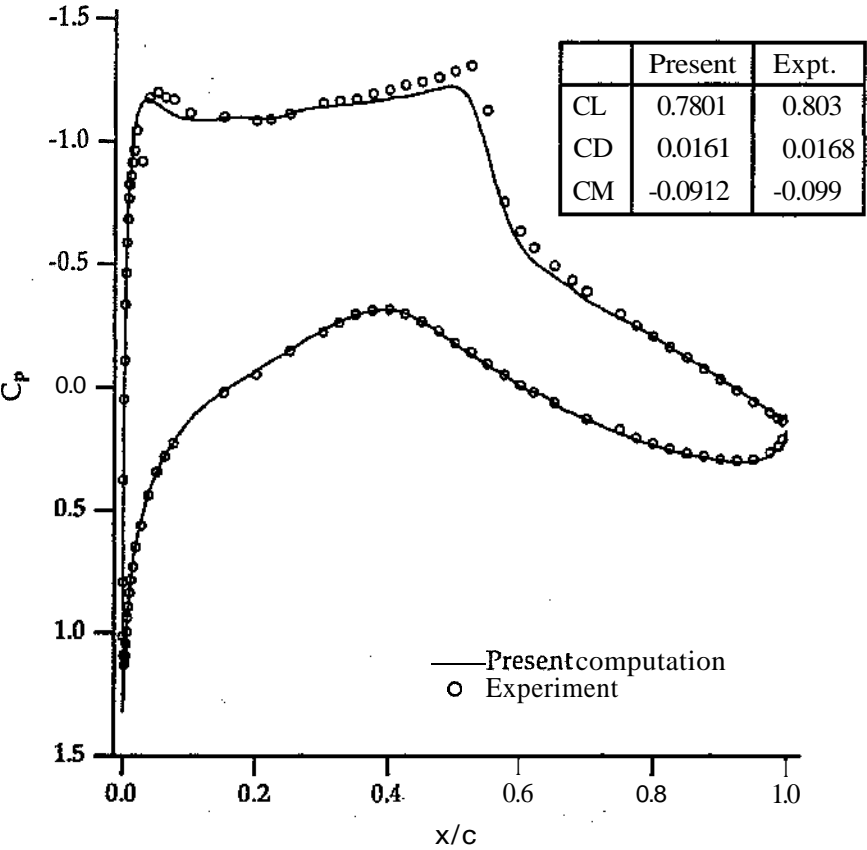


Figure 2. Surface pressure distribution for an RAE-2822 airfoil.
($M_\infty = 0.73$, $\alpha = 2.79^\circ$, $Re = 6.5 \times 10^6$, Grid = 257×62)

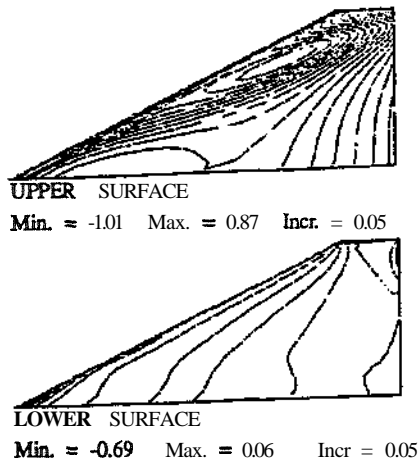


Figure 3. Lines of constant C_p on the wing surface.
($M_\infty = 0.85$, $\alpha = 10^\circ$, $Re_\infty = 2.38 \times 10^6$)

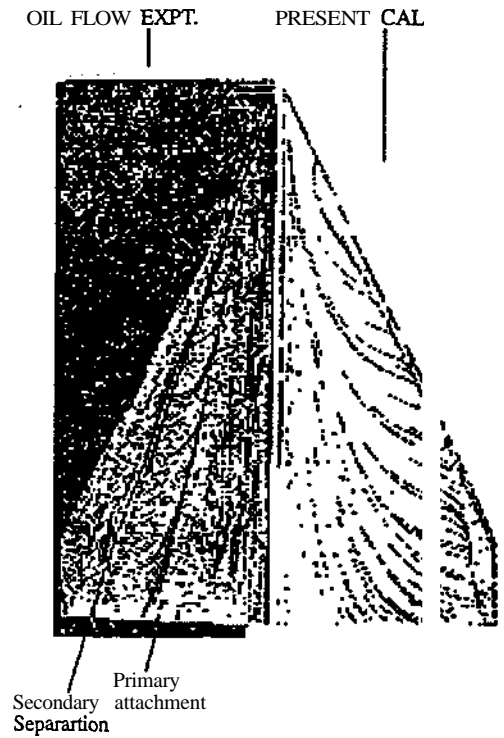


Figure 4. Comparison of skin-friction line with oil flow experiment

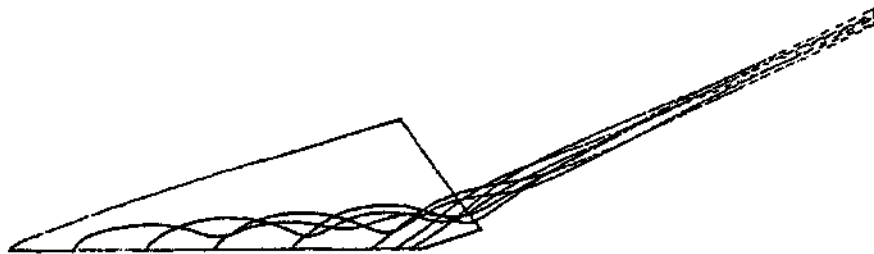


Figure 5. Spiralling streamlines from the leading edge

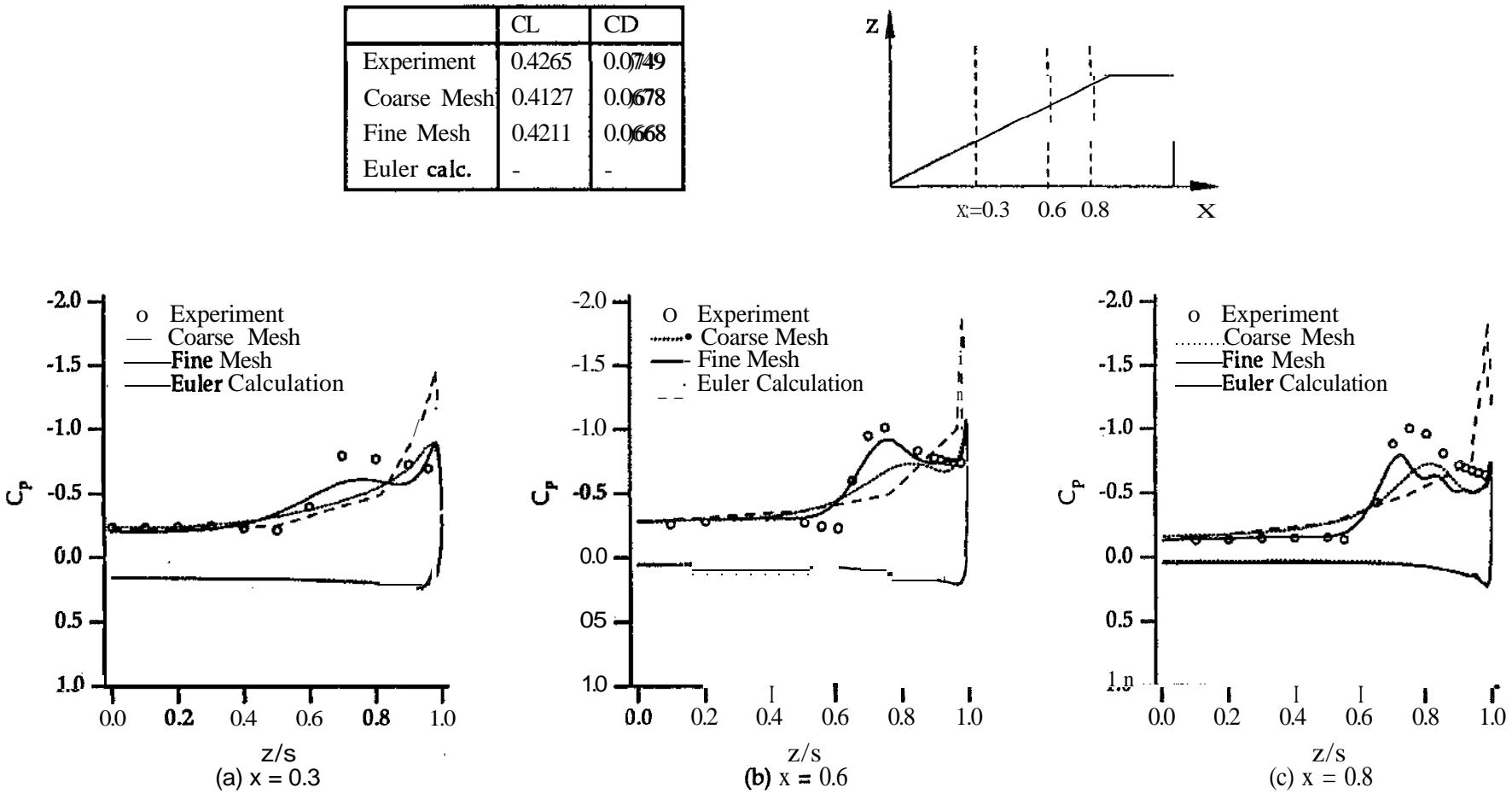


Figure 6. Spanwise surface pressure distributions, at three streamwise locations, for a 65° cropped delta wing with rounded leading edge. ($M_\infty = 0.85$, $\alpha = 10^\circ$, $Re = 2.38 \times 10^6$)